CFD APPROACH AS DESIGN OPTIMIZATION FOR GAS TURBINE TUBULAR COMBUSTOR

H. A. Bhimgade¹, C. A. Mahatme², P. S. Barve and N. D. Gedam

Assistant Professor, Department of Mechanical Engineering, YCCE, Nagpur (India)

{E-mail: ¹bhimgade.harshu5@gmail.com, ²chetanmahatme@gmail.com}

Abstract

Gas Turbine Combustor mixing processes are of paramount importance in the combustion and dilution zones. In the primary zone of combustor, good mixing is essential for high burning rates and to minimize soot and nitric oxide formation, whereas the attainment of a satisfactory temperature distribution (pattern factor) in the exhaust gases is very dependent on the degree of mixing between air and combustion products in the dilution zone. A primary objective of combustor design is to achieve satisfactory mixing within the liner and a stable flow pattern throughout the entire combustor, with no parasitic losses and with minimal length and pressure loss. Accordingly, in present study an attempt has been made through Computational Fluid Dynamics (CFD) approach using CFX 13.0 to analyze the flow patterns within the combustion liner and through different air admission holes, namely, primary zone, intermediate zone, dilution zone and from these the temperature distribution in the liner and at walls as well as the temperature quality at the exit of the combustion chamber is obtained for tubular combustion chamber designed for gas turbine engine. The aim is to illustrate what can be done and also to identify trends and those areas where further work is needed.

Keywords: Computational Fluid Dynamics, Flowfield, Gas Turbine Tubular Combustor

1. Introduction

Gas turbine combustor designs are becoming increasingly challenging in order to meet the stringent requirements such as lower maximum exit temperatures, lower emissions, higher durability, lower fabrication and maintenance cost and reduced design and time-to-market cycle times. These requirements necessitate more emphasis on Computational Fluid Dynamics (CFD) simulation of the combustion flowfield to reduce testing and improve performance which is a complex problem. The flowfield conditions at the combustor exit in real gas turbine engines are highly non-uniform in temperature, pressure, and velocity. These non-uniformities are a function of the combustion chamber flow arrangement and geometry. The major features of these combustion chambers include the chamber liners, entrance swirlers, fuel nozzles, liner cooling slots and holes, as well as primary and secondary dilution holes. One major consideration in the designing of a combustor is

the division of the inlet mass flow into separate flow paths. A large portion of the combustor inlet flow is used for cooling the metal casing and diluting the combustion products. Several experimental and numerical studies have been performed to analyze and model the combustor flows. According to Snyder et al. [1] used the advanced computational analysis system to optimize the combustor exit temperature in the Pratt & Whitney PW6000 engine. They successfully used this computational design tool to tailor combustor exit temperature profiles to within design limits by performing a parametric study of dilution hole patterns. Although, no information with respect to the hole pattern geometry was given, the optimized dilution hole pattern was incorporated into the PW6000 engine to improve durability and prolong turbine life indicating the computational effort was successful. B. Zamuner et al. [2] carried out a study of

numerical simulation of reactive flow in a tubular gas turbine combustor with detailed kinetic effects and concluded that, Flame structures, combustion regimes and pollutants emission are often difficult to predict in aircraft combustors because they are closely related to the turbulent nature of the flow and requires a complete iterative procedure to improve the predicted values. Sierra et al. [3] carried out work upon the combustion chamber was part of a 70MW gas turbine used in an operating combined cycle power station and concluded that the range of pressure imbalance of primary air applied in the refined model was adopted and this effect is interpreted as a flame expansion process caused by inlet air pressure variations. Fureby et al. [4] carried out an experimental and computational study of a multi-swirl gas turbine combustor. The computational approach pursued was large eddy simulation (LES), which provides a compromise between accuracy and cost. LES attempts to capture the dynamics and evolution of the large-scale flow, at the same time as it allows for inclusion of realistic flow and chemistry parameters. The flow was affected by the exothermicity through the volumetric expansion, increased molecular viscosity and baroclinic torque, resulting in flow acceleration downstream of the flame, and the development of wall jets. Di Martin et al. [5] studied a reactive CFD analysis of complete annular gas turbine combustor module for aero engines application. All the complex combustor features like cooling holes, primary and dilution holes, swirler, fuel injector etc. were considered and fully coupled into the CFD calculation. The internal combustor flow field of mixture were studied and observed that effusion holes, which were very small in diameter and very large in number was not meshed but the effect of drilled liners was properly modeled by means of source terms and well tested correlations that link holes mass flow rate with the pressure drop across the liners. Massimo Masi et al. [6] studied the numerical and experimental analysis of the temperature distribution in a hydrogen fuelled combustor for a 10 MW gas turbine as part of a ministerial project coordinated by ENEL Ricerca. The computed temperature profiles showed acceptable matching with

measurements in the liner wall and cost CFD model presented was able to capture the mean temperature field within the combustor, which was the fundamental requirement for the planned thermoacoustic characterization of the analyzed combustor. Daero Joung and Kang [7] designed and developed a small size gas turbine combustor of a reverse flow, semi-silo type for power generation operated in a lean premixed mode to achieve stable combustion. In this experiment, combustor with multi swirler (main swirler and pilot swirler at inlet of burner), annular nozzle having, pilot fuel and main fuel injector is simulated. The premixed coherent flame model (PCFM) is applied for partially premixed methane/air with an imposed downstream flame area density (FAD) to avoid flashback and incomplete combustion. The combustion efficiency reached about 99.9% with higher inlet temperature. Channiwala & Kulshreshtha [8] the paper presented three dimensional model investigated by numerically to study the flow behavior in pre-diffuser, dump region, liner, inner and outer annuli and swirler exit.

In this work, an attempt has been made through reacting CFD analysis to achieve the proper mixing of combustion products in the dilution zone and exit temperature uniformity. During this process, it has been ensured that the total pressure remains almost the same. Analysis has been carried out using CFX 13.0, for tubular gas turbine combustor without casing and, swirler and fuel injector at inlet. Results obtained through computation shows proper mixing of combustion products with the admission air through different zone holes and almost uniform temperature at exit.

2. Computational model

2.1 Combustor geometry

In this study, combustor geometry for analysis is a tubular combustor having an axial flow swirler with 8 aerofoil shape vanes at the inlet of combustor provided to maximize the inlet air turbulence and the injector with 6 holes of 250 micron diameter is provided at the entrance of the primary zone. The length of the liner and

number of holes on the liner are designed accordingly and all the dimensions are provided in the Table 5.2.1 and Table 5.2.2. The Fig. 5.1 shows the cross sectional view of the combustor which is designed according to the design methodology proposed by Lefebvre A. H. and Ballal D. R [9].

2.2 Combustor mesh and boundary condition

The tetra-hedron unstructured grid has been generated using GAMBIT 2.2. The Fig. 5.2 shows the 3-D grid model of tubular combustion chamber with 148808 nodes and 781989 elements selected for CFD simulation. Total pressure and total temperature have been specified at the combustor inlet, static pressure along with the target mass flow rate has been specified at the combustor core exit. Turbulent intensity and hydraulic diameter have been specified as the initial conditions for the inlet turbulence. At the combustor bleeds where no combustion takes place mass flow rate boundary condition has been specified in such a way that the prescribed quantity of flow goes out of the domain. All the combustor walls have been treated to be adiabatic. The Fig. 5.3 shows the computational domain.

2.3 Governing equations

It exists six equations that should be solved to model the flow field. These equations are continuity, momentum, energy, species transport, turbulence and combustion equations. In the present study, flow is treated to be steady, turbulent, compressible and reacting. The governing Navier-Stokes equations (RANS) for the conservation of mass, momentum, energy and species concentration for the gas, together with an equation of state are approximated for each mesh cell. The resulting sets of equations are solved numerically to obtain the flow field, mixing and combustion data. The following Table-3 shows the computational model for the combustor analysis.

3. Flow solution

ANSYS CFX v13.0 is used as solver. The numerical settings for the solver are described below.

3.1 Time stepping

The problem is solved as a steady state flow problem, consistent with the RANS turbulence modelling used, which means that relatively large time steps are used in order to achieve a converged solution as quickly as possible.

3.2 Heat transfer

“Thermal energy” model is used, which means that the total energy models the transport of enthalpy including the kinetic energy effects. This model should be used where there is change in density or the Mach number exceeds 0.2; in both of these cases kinetic energy effects are significant. In ANSYS CFX, when one chooses thermal energy the fluid is modelled as compressible, regardless of the original fluid condition, i.e. gases with Mach number less than 0.2. One should know that incompressible fluid does not exist in reality but for the gases with Mach number less than 0.2 the compressible effects are in general negligible.

3.3 Turbulence

For the turbulence k- ϵ turbulence models are used. The k- ϵ model is one of the most common turbulence models. It is a two equation model that includes two extra transport equations to represent the turbulent properties of the flow. This allows the model to account for history effects like convection and diffusion of turbulent energy.

3.4 Combustion model

The laminar flamelet model is used as combustion model which solves only two transport equations for a large number of species (low computational cost). It provides information on minor species and radicals (such as CO and OH). As well as accounting for turbulent fluctuations in composition (presumed PDF), it models local extinction at high scalar dissipation rates or shear strain. The model is only applicable for two-feed systems (fuel and oxidizer), and requires a chemistry library as input. Diesel fuel library is generated using CFX-RIF. Diesel fuel, modeled as a two-

component surrogate fuel (by mass 62.44% n-C₁₀H₂₂ and 37.56% A₂CH₃-C₁₁H₁₀) [12]. The same pressure level must apply to the whole domain and the model is only for non-premixed systems.

3.5 Convergence criteria

In order to determine if convergence is obtained, residuals are constantly monitored and when they are reasonable flattened out, the run is stopped and the results are post-processed.

4. Discussion

In this case, the incoming air at the pressure and temperature enters into the combustion chamber and through liner holes, reacts with the atomized fuel. The effect of providing aerofoil swirler at the inlet on flowfield and on the combustor performance is discussed below:

Fig. 5.4 shows the velocity distributions at radial locations of the combustion chamber. Velocity, pressure and temperature measurements are carried out at centerline of combustion chamber and in the radial direction, at $r/R = 0.35$. In this case, the low velocities are encountered in the primary zone at axial as well as radial locations. These low velocities are

beneficial for both combustion stability and mixing. The fact is evident from Fig. 5.5, which shows the flame stability. Good mixing and the recirculation is observed at central core of primary zone, which may offer stable narrow flame. The velocity levels are slightly higher in the dilution zone compared to the primary zone. As more air enters through the dilution zone, the velocity levels increases. This may be due to the fact that pressure drop is manifested in the increased velocity levels for cold flow studies. This pressure drop is graphically represented in Fig. 5.7. Higher pressure drop is witnessed in the dilution zone which leads to higher velocities near the exit of the combustion chamber.

The velocity contour for reacting flow in radial direction are shown in the Fig. 5.4. It is observed that in the primary zone of the combustion chamber, the velocities of flow are lower near the wall and at the central a small size recirculation zone occurred. The lower velocities are beneficial for the combustion stability and small size recirculation zone which is beneficial for the temperature distribution at the exit of the combustion chamber, narrow flame and good mixing.

Table 1 Dimensions of various parts of combustor

Sr. No.	Part	Dimensions
1	Hub Diameter, D_{mb}	0.18 m
2	Swirl Diameter, D_{mb}	0.3 m
3	Swirl No., S_N	0.8
4	D_{nb}/D_{nw}	0.6
5	Liner length, L_L	0.45 m
4	Liner Diameter, D_L	0.15 m

Table 2 Number of holes (n) and hole diameter (d_h) on the Liner

Chamber	Liner area m^2	Air admission holes in primary zone				Air admission holes in dilution zone					
		n	dh (m)	n	dh (m)	n	dh (m)	n	dh (m)	n	dh (m)
Tubular	0.0079	8	0.004	24	0.010	3	0.03	24	0.004	3	0.03

The Fig.1 shows the cross sectional view of the combustor which is designed according to the design methodology proposed by Lefebvre A. H. and Ballal D. R [9].

Fig.1 : Cross-sectional view of combustor

2.2 Combustor mesh and boundary condition

The tetra-hedron unstructured grid has been generated using GAMBIT 2.2. The Fig. 2 shows the 3-D grid model of tubular combustion chamber with 148808 nodes and 781989 elements selected for CFD simulation.

Fig.2 : 3-D grid model

Total pressure and total temperature have been specified at the combustor inlet, static pressure along with the target mass flow rate has been specified at the combustor core exit. Turbulent intensity and hydraulic diameter have been specified as the initial conditions for the inlet

turbulence. At the combustor bleeds where no combustion takes place mass flow rate boundary condition has been specified in such a way that the prescribed quantity of flow goes out of the domain. All the combustor walls have been treated to be adiabatic. The Fig. 3 shows the computational domain.

2.3 Governing equations

Fig. 3 : Computational domain

It exists six equations that should be solved to model the flow field. These equations are continuity, momentum, energy, species transport, turbulence and combustion equations. In the present study, flow is treated to be steady, turbulent, compressible and reacting. The governing Navier-Stokes equations (RANS) for the conservation of mass,

momentum, energy and species concentration for the gas, together with an equation of state are approximated for each mesh cell. The resulting sets of equations are solved numerically to obtain the flow field, mixing and combustion data. The following Table 3 shows the computational model for the combustor analysis.

Table 3 : Computational mode

Fluid model	Thermal enegy
Turbulence model	k- ϵ
Combustion model	Laminar flamelet with PDF
Radiation model	Discrete transfer
Combustion reaction	Flamelet library
Nitrogen	Constraint

3. Flow solution

ANSYS CFX v13.0 is used as solver. The numerical settings for the solver are described below.

3.1 Time stepping

The problem is solved as a steady state flow problem, consistent with the RANS turbulence modelling used, which means that relatively large time steps are used in order to achieve a converged solution as quickly as possible.

3.2 Heat transfer

“Thermal energy” model is used, which means that the total energy models the transport of enthalpy including the kinetic energy effects. This model should be used where there is change in density or the Mach number exceeds 0.2; in both of these cases kinetic energy effects are significant. In ANSYS CFX, when one chooses thermal energy the fluid is modelled as compressible, regardless of the original fluid condition, i.e. gases with Mach number less than

0.2. One should know that incompressible fluid does not exist in reality but for the gases with Mach number less than 0.2 the compressible effects are in general negligible.

3.5: Convergence criteria

In order to determine if convergence is obtained, residuals are constantly monitored and when they are reasonably flattened out, the run is stopped and the results are post-processed.

4. Discussion

In this case, the incoming air at the pressure and temperature enters into the combustion chamber and through liner holes, reacts with the atomized fuel. The effect of providing aerofoil swirler at the inlet on flowfield and on the combustor performance is discussed below:

Fig. 4 shows the velocity distributions at radial locations of the combustion chamber. Velocity, pressure and temperature measurements are carried out at centerline of combustion chamber and in the radial direction, at $r/R = 0.35$.

Fig. 4 : Radial velocity contour

In this case, the low velocities are encountered in the primary zone at axial as well as radial locations. These low velocities are beneficial for both combustion stability and mixing. The fact is evident from Fig. 5, which shows the flame stability. Good mixing and the recirculation is observed at central core of primary zone, which may offer stable narrow flame.

The velocity levels are slightly higher in the dilution zone compared to the primary zone. As more air enters through the dilution zone, the velocity levels increases. This may be due to the fact that pressure drop is manifested in the increased velocity levels for cold flow studies.

Fig. 5 : Axial temperature contour

This pressure drop is graphically represented in Fig. 6. Higher pressure drop is witnessed in the dilution zone which leads to higher velocities near the exit of the combustion chamber.

Fig. 6 : Pressure distribution for reacting flow

The velocity contour for reacting flow in radial direction are shown in the Fig. 7. It is observed that in the primary zone of the combustion chamber, the velocities of flow are lower near the wall and at the central a small size recirculation zone occurred. The lower velocities are beneficial for the combustion stability and small size recirculation zone which is beneficial for the temperature distribution at the exit of the combustion chamber, narrow flame and good mixing.

In Fig. 8 shows a case of axial CFD result, higher temperature found initially which is beneficial for the complete combustion. Lower temperature at exit of the combustion chamber is beneficial for turbine blade life and nozzle life. Axial and radial CFD temperature distributions are quite differ from each other but does not affect at exit of the combustion chamber.

Fig. 7 : Velocity distribution for reacting flow

Fig. 8 : Temperature distribution for reacting flow

6. Conclusion

The design of tubular or can type gas turbine combustion chamber is carried out using diesel (surrogate) as fuel and the design is then validated using Numerical Approach. The results of numerical approach of present study compared with the previous experimental work done by Channiwala et al. It has been found the close relationship between results. Qualitative and quantitative agreement of CFD results suggests that the basic assumptions and boundary conditions as well as the problem definition for CFD analysis can be applied to understand the flow phenomena, temperature contours and air flow distribution for combustion chamber. The almost consistent centerline temperatures achieved along the centerline of combustion chamber validates the design methodology proposed and presented in this paper. The maximum centerline temperature recorded by CFD simulation is in the vicinity of 2100 K. The pressure loss along the combustion chamber is 10% of the inlet pressure. The velocity profiles show an increasing trend along the length of combustion chamber, but low velocities are encountered in primary zone which is beneficial for combustion stability.

References

- [1] T. Snyder, J. Stewart, M. Stoner & R McKinney, Application of an Advanced CFD Based Analysis System to the PW6000 Combustor to Optimize Exit Temperature Distribution – Part II: Comparison of Predictions to Full Annular Rig Test Data, ASME Paper No. 2001-GT-0064.
- [2] B. Zamuner, B. Bourasseau, C. Berat, H. Niemann, Gas Turbine Engine Combustion, Emissions and Alternative Fuels, RTO AVT Symposium held in Lisbon, Portugal, 12-16 October 1998, and published in RTO MP-14.
- [3] F.Z. Sierra., J. Kubiak, G. Gonzalez, G. Urquiza, Prediction of temperature front in a gas turbine combustion chamber, Applied thermal engineering, vol. 25, 2005, issues 8-9, p. 1127-1140.
- [4] C. Fureby, F.F. Grinstein, G. Li, E.J. Gutmark, An experimental and computational study of a multi-swirl gas turbine combustor, Proceedings of the Combustion Institute, vol. 31, 2007, p. 3107–3114.
- [5] Di P. Martino, G. Cinque, A. Terlizzi, G. Mainiero, S. Colantuoni, Reactive CFD Analysis in a Complete Combustor Module for Aero Engines Application, Universita Degli Studi Di Napoli, Federico II, April 26-28, 2009.
- [6] P. Gobbato, M. Masi, A. Toffolo, A. Lazzaretto, Numerical simulation of a hydrogen fuelled gas turbine combustor, International journal of hydrogen energy, 36, 2011, p. 7993 -8002.
- [7] Daero Joung, Kang Y. Huh, An. Yunho, Parametric simulation of turbulent reacting flow and emissions in a lean premixed reverse flow type gas turbine combustor, Journal of engineering for gas turbines and powers, vol. 34, 2012, 021501.
- [8] S. A. Channiwala & D. Kulshreshtha, Numerical Simulation Approach As Design Optimization For Micro Combustion Chambers, Proceedings of ICFD 10, Tenth International Congress of Fluid Dynamics, December 16-19, 2010.
- [9] A. H Lefebvre. & D. R. Ballal, Gas turbine combustion (Alternative fuels and emissions), Taylor and Francis group, Third edition, 2010.
- [10] A. Khodabandeh, CFD Modelling Combustor of Generic Gas Turbine, *Master's Thesis in Solid and Division of Fluid Dynamics* Department of Applied Mechanics, Chalmers University of Technology, Göteborg, Sweden 2011.
- [11] Ahmed Abed Al-Kadhem Majhool, Advanced Spray and Combustion

Modeling, University of Manchester, Faculty of Engineering and Physical Sciences, 2011.

- [12] C. Fureby, F.F. Grinstein, G. Li, E.J. Gutmark, An experimental and computational study of a multi-swirl gas turbine combustor, Proceedings of the Combustion Institute, vol. 31, 2007, p. 3107–3114.
- [13] A. Bicen, D. Tse & J. Whitelaw, Flow and Combustion Characteristics of an Annular Combustor, *Combustion and Flame*, vol. 72, 1988, p. 175-192.